International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 9, Issue 3, Jun 2019, 607-612


# COMPUTATIONAL ANALYSIS OF FLOW OVER BLUFF BODIES USING AERO SPIKES

**A. ADAIKALARAJ[1], DR. M. SUNDARARAJ[2] & M. SYEDHALEEM[1]**

*[1]Assistant Professor, Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Tamil Nadu, India*

*[2]Head and Associate Professor, Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Tamil Nadu, India*

***ABSTRACT***

***This paper presents the computational analysis of the effect of aero spikes on bluff bodies. In the present study, a blunt nosed model was designed with and without spike. Two different shaped spikes were chosen for our study. Efforts were taken to obtain the flow pattern and the pressure distribution field over the nose of the blunt body. Flow pattern was observed and recorded for different shaped spike attached in-front of the body. A comparative study was also done to observe which shape of the aero spike provides a better zone of the pressure field. It is observed that the radius of curvature of the streamline gets reduced by aero spike. Also, there occurs a deflection in the streamline which in turn resulted in creating a narrow zone of the positive pressure region. The zone of influence gets reduced by hemispherical shaped spike. The models were designed using CATIA V5 software and the analysis were carried out using ANSYS software packages.***

***KEYWORDS:** Aero Spike, Pressure Zone, Blunt Nosed Body & Vortex*

**Received:** Mar 14, 2019; **Accepted:** Apr 04, 2019; **Published:** May 05, 2019; **Paper Id.:** IJMPERDJUN201965

Original Article

## INTRODUCTION

In low speed and high speed applications, the drag reduction in blunt-nose body plays an important role. The formation of high drag in hypersonic flying objects is because of the blunt nose region. This blunt region leads to high pressure zone at the front region of the nose. But we could not change the blunt shape of the vehicles since this shaped nose act as the safeguard of the body during an aerodynamic heating condition. Secondary objects were fixed in front of the blunt nose region projecting outside to reduce drag. That projections (secondary objects) were termed as Aero spikes and this is accepted as the effective method of drag reduction. These Aerospikes actually shifts the forward stagnation point away from the nose region. Because of this change in the location of stagnation point, the extent of the pressure zone comes down which in turn resulted in drag reduction. The flow process related to the shifting of the stagnation point in front of the blunt nose will results in reducing the strength of the detached shock, and reduces the extension of the pressure zone influenced by the detached shock. Many experimental studies have been conducted to analyze the frontal body flow field of the spiked blunt body. This present work is an attempt to measure the positive zone of influence, and the pressure field over the frontal region of a blunt-nosed body, with and without spike. To study this the flow pattern over the desired blunt-nosed body, with without spikes, was analyzed and visualized using ANSYS software packages, at a specific Reynolds number. The Computational analysis is one of the cheap and best techniques which can be used for the

visualization of real flow problems. William H. Bettes et al., presented a paper. In his paper, the drag coefficient on the automobiles can be reduced by 24% by changing the front body design shapes and 11% by changing rear body design of the vehicle which saves 25 percent fuels consumption. Edwin J. Saltzman et al. conducted experimental studies on blunt bodies. From their experiments, the rounded corner vehicle when compared with square shaped configurations shows that reduction in aerodynamic drag of about 40 percent can be achieved when compared to the square shaped configuration. Ashish Vashishtha, et al., made an attempt to study the comparative results between drag on the blunt nosed body with and without breathing nose at the Mach 1.96. The breathing nose resulted in less drag because of the manipulation of the high pressure at the nose and the low pressure at the base simultaneously. A maximum drag reduction of 21% was obtained. This drag reduction is because of the combined effect of decrease of high pressure at the nose and increase of low pressure at the base. In this work, we are considering the projection technique in terms of aero spike to reduce the drag.

## MODELLING AND SIMULATIONS

The model was designed using CATIA V5 software. In the present study the flow around a blunt body without spike, blunt body with spike of hemispherical end and a blunt body with spike of arrow end were designed and analyzed. The models were shown in figure 1. The overall length of the blunt body was considered as 90mm and the width of the blunt body was taken as 60mm. The nose is having a diameter of 60mm. The analyses were further extended by changing the velocities as 50m/s, 60m/s and 70m/s.

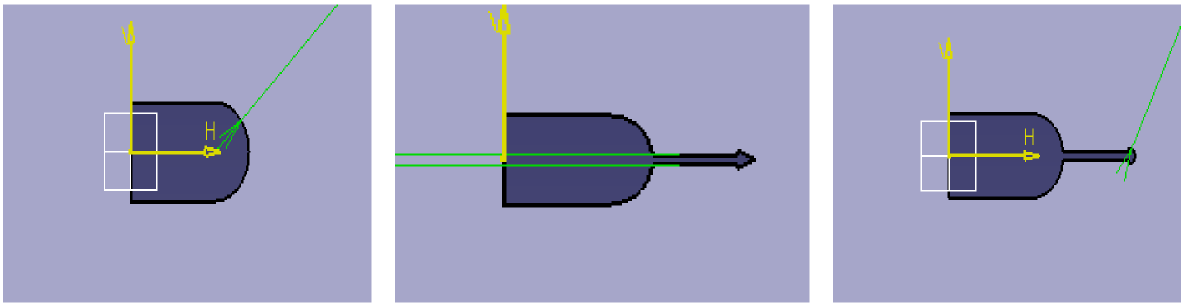


**Figure 1: Blunt Body with Various Aero Spike Configuration**

Figure 2 shows the discretized model of the blunt bodies with aero spikes. The domain was created around the model since analyses deals with external aerodynamics. The models designed using CATIA V5 were imported to ICEMCFD for meshing and the discretized model was shown below. Meshing was done finely near the areas where we need to capture the flow field. Unstructured mesh was selected for this meshing process. After the completion of discretization stage, the model was imported for analysis purposes.

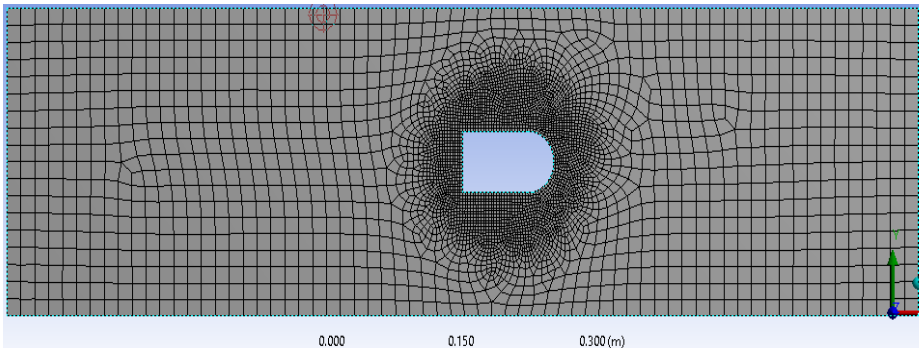


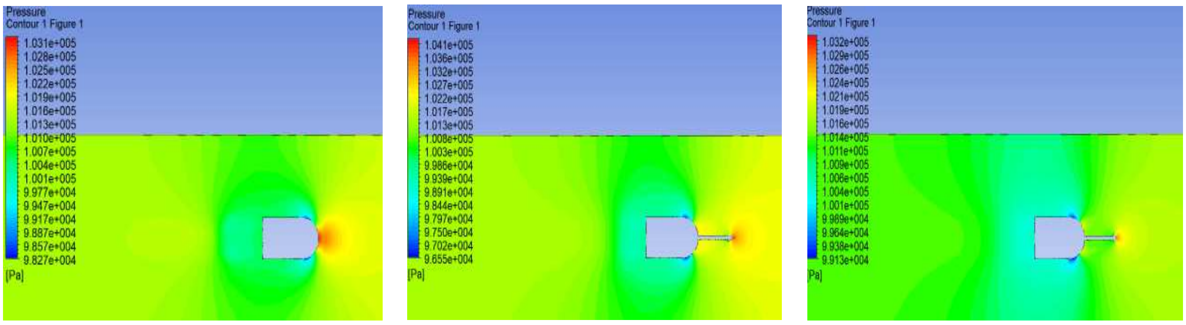

**Figure 2: Discretized Model**

## RESULTS AND DISCUSSIONS

Three models were chosen for our analysis. Base without projection, base with sharp edge projection and base with hemispherical projection were chosen for our study. Air is allowed to flow over the different models at three different velocities. Boundary conditions were given and the analyses were carried out for three different configurations. The results were observed and recorded in this paper. The obtained results were compared with the base shape without aero spikes. In order to reduce the drag over these kind of blunt bodies, we need to reduce the positive pressure at the nose region. By placing the aero spike in front of the nose region we could reduce the positive pressure field region over the nose area.

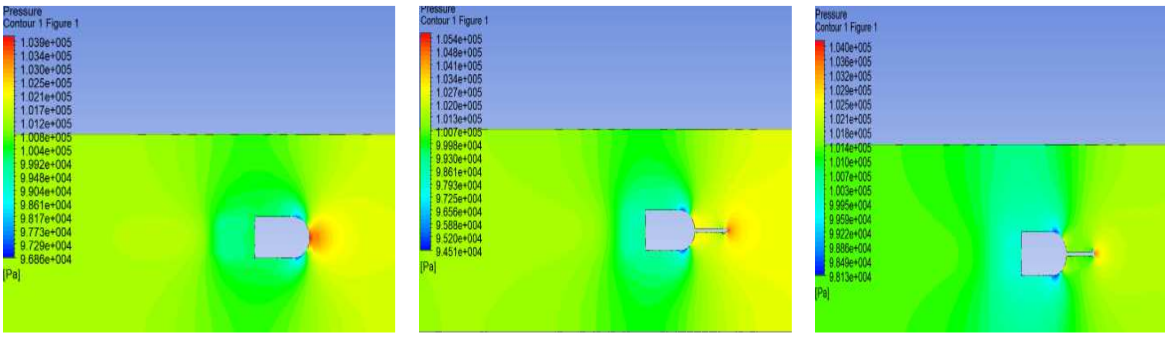
### Pressure Contours

Pressure Contour at Velocity V= 50m/s



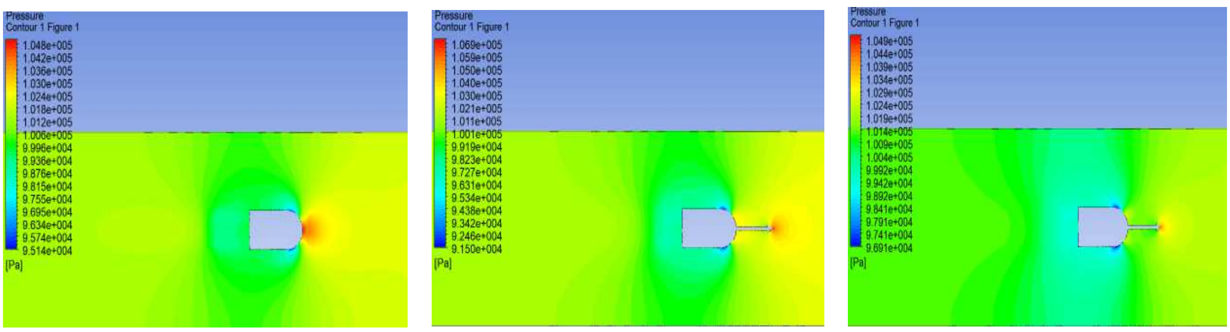
**Without Spike** **With Arrow Spike** **With Hemispherical Spike**

Pressure Contour at Velocity V= 60 m/s



**Without Spike** **With Arrow Spike** **With Hemispherical Spike**

Pressure Contour at Velocity V= 70m/s

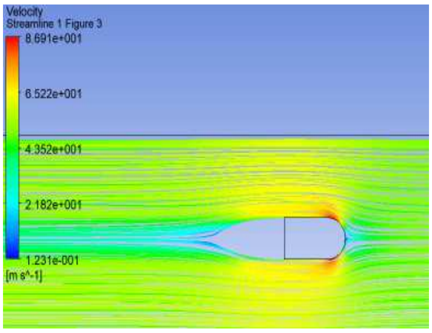


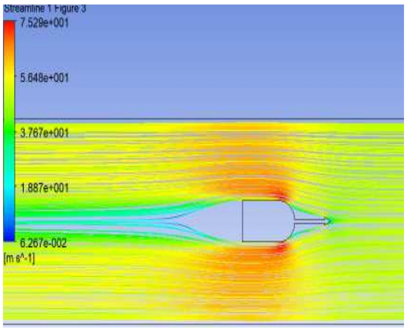
**Without Spike** **With Arrow Spike** **With Hemispherical Spike**

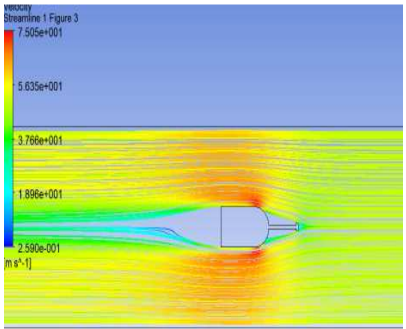
**Figure 3: Pressure Contours for Various Configurations at Different Velocities**

**Velocity Streamline Contour**

Velocity Streamline Contour at V= 50m/s

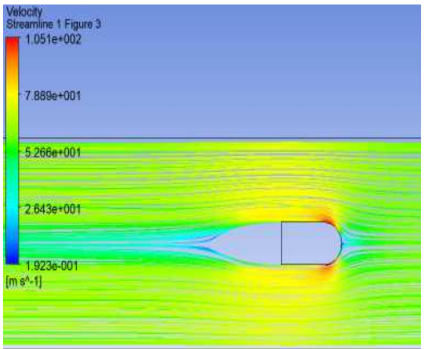


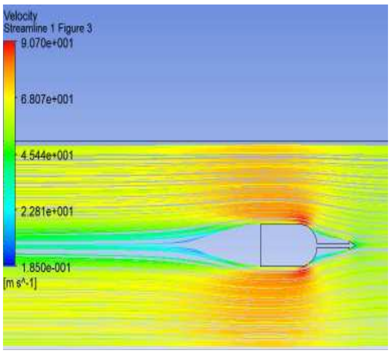



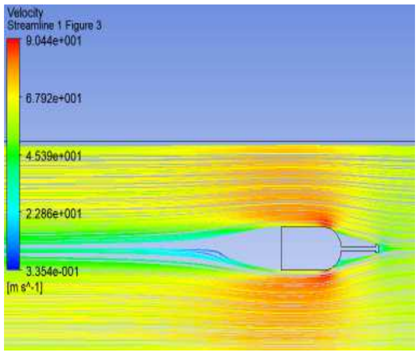




**Without Spike** **With Arrow Spike** **With Hemispherical Spike**

Velocity Streamline Contour at V= 60m/s

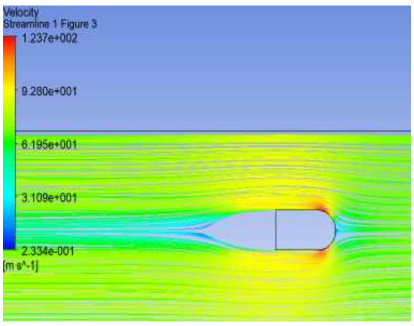


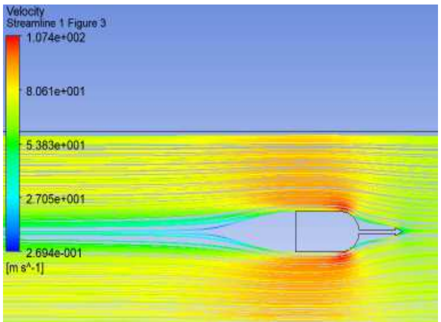



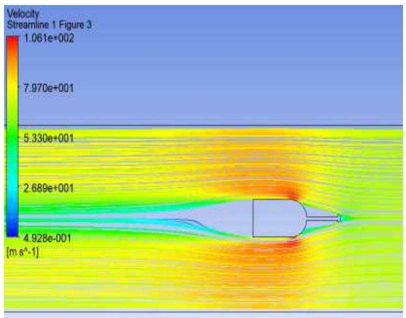




**Without Spike** **With Arrow Spike** **With Hemispherical Spike**

Velocity Streamline Contour at V= 70m/s



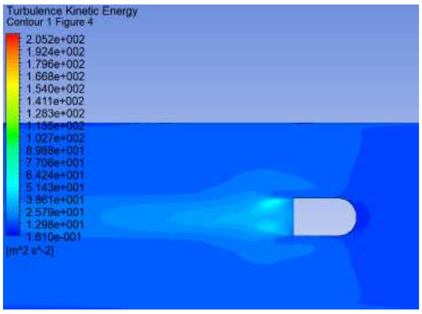




**Without Spike** **With Arrow Spike** **With Hemispherical Spike**

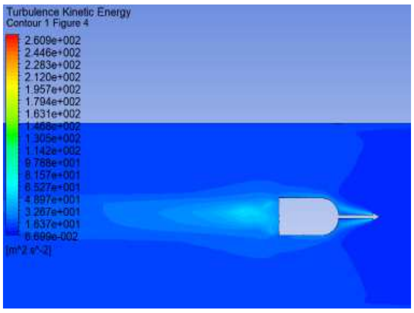
**Figure 4: Velocity Streamline Contour for Different Configurations at Various Velocities**

**Turbulence Kinetic Energy Contour**

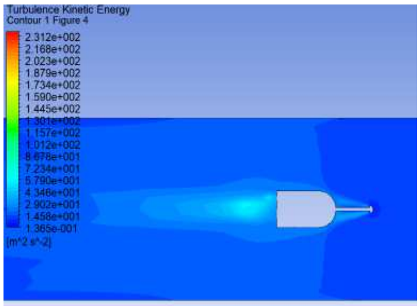
Turbulence Kinetic Energy Contour at Velocity V=50m/s



**Without Spike**

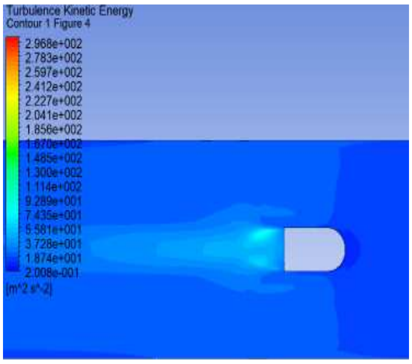


**With Arrow Spike**

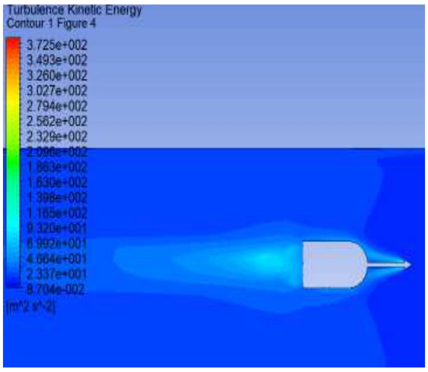


**With Hemispherical Spike**

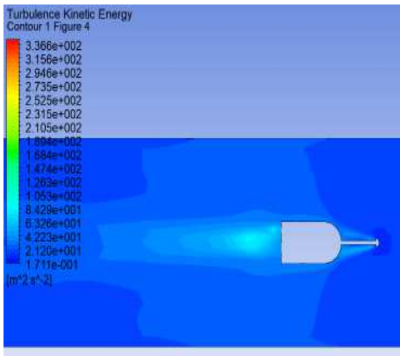
Turbulence Kinetic Energy Contour at Velocity V=60m/s



**Without Spike**

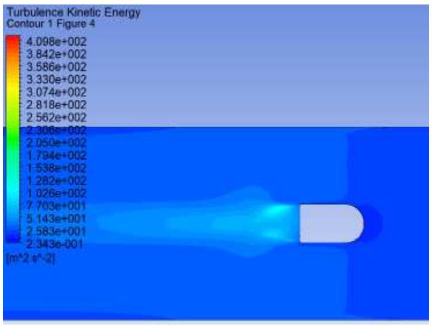


**With Arrow Spike**

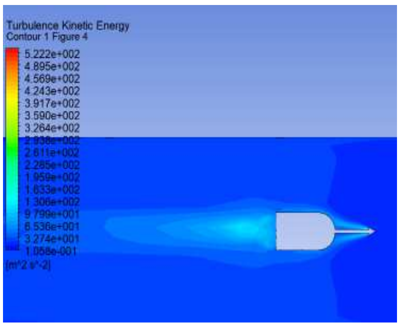


**With Hemispherical Spike**

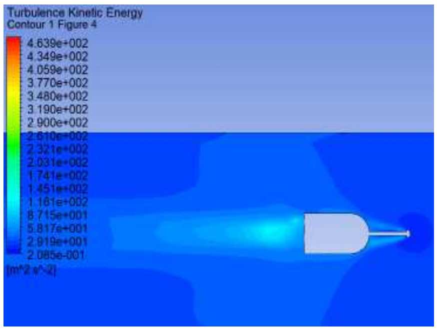
Turbulence Kinetic Energy Contour at Velocity V=70m/s



**Without Spike**



**With Arrow Spike**



**With Hemispherical Spike**

**Figure 5: Turbulent Kinetic Energy Contour for Different Configurations at Various Velocities**

## CONCLUSIONS

The analyzed results were summarized in this paper. When compared with all the three configurations, the spike with hemispherical configurations attached to the basic blunt nosed body shows much influence on the flow field, twin vortex formation and reverse flow distance. There shows a considerable increment in the reverse flow distance for model with and without spike. It is observed that the hemispherical spike has given the lowest vortex length compared to the basic blunt nosed body. Whereas coming to the two different configuration of hemispherical spike L/W=1 and 1.5, it is observed that the effect on the flow field is insignificant. It is found that the spike reduces the radius of curvature of the approaching

streamline, and the deflection of the streamline is towards the shoulder of the basic body resulting in a narrow zone of positive pressure hill at the nose. The spike with hemispherical nose reduces the zone of influence. So finally it can be concluded that choosing the spike configuration L/W = 1 will be beneficial as the economic concern.

***REFERENCES***

1. *William H. Bettes "The aerodynamic drag reduction of road vehicles in earlier" Journal Of Engineering & Science / January 1982*
2. *Edwin j, Saltman and Robert R. Meyer "drag reduction obtained by rounding vertical corners of box shaped vehicle" NASA Flight research center march 1974.*
3. *Ashish Vashishtha, Hemant Sharma, P. Lovaraju, and E. Rathakrishnan "Breathing Blunt Nose Concept for Drag Reduction in Supersonic Flow"*
4. *E. Rathakrishnan "Effect Of Splitter Plate On Bluff Body Drag" Aiaa Journal Vol. 37, No.9,September 1999*
5. *Khalid M. Sowed and E. Rathakrishnan "Front Body Effects On Drag And Flow Field Of A 3-D Bluff Body" AIAA Journal, VOL.31,NO 7;technical notes*
6. *G. K. Suryanarayana, Hemming Pauer, G. E. A. Meier "Bluff-Body Drag Reduction By Passive Ventilation" Journal of Experiments in Fluids16, 73-81 (1993).*
7. *Dwivedi, Y. D., & Bhargava, V. (2018). Prediction of Aerodynamic Characteristics for Slender Bluff Bodies with Different Nose Cone Shapes. Available at SSRN 3124819.*
8. *Stephen A. Whitmore, Stephanie Sprague, Jonathan W. Naughton "Drag Reduction Using Fore Body Surface Roughness" NASA/TM-2001-210390*
9. *Shashank Khurana, Kojiro Suzuki, Ethirajan Rathakrishnan, "Flow Field around a Blunt-nosed Body with Spike, International Journal of Turbo & Jet-Engines, Volume 29, Issue 4,Dec 2012,pp 217-221.*
10. *Reddy, P. R., & Saikiran, M. (2016). Aerodynamic Analysis of Return Channel Vanes in Centrifugal Compressors. International Journal of Mechanical Engineering (IJME), 5(1), 73-82.*
11. *R. Kalimuthu and E. Rathakrishnan, Aerodynamic Characteristics of Blunt Nosed Body with and without Aerospike at Mach 5.87,Proceedings of the 10th Asian Symposium on Visualization, 2010*
12. *V.L. Zhdanov, H.D. Papenfuss "Bluff Body Drag Control By Boundary Layer Disturbances" Journal of Experiments in Fluids 34 (2003)460–466*
13. *Sang-Joon Lee, Sang-Ik Lee, Cheol-Woo Park "Reducing The Drag On A Circular Cylinder By Upstream Installation Of A Small Control Rod" Journal of Fluid Dynamics Research 34 (2004) 233–250.*
14. *Yoshio Yajima, Osamu Sano "A note on the drag reduction of a circular cylinder due to double rows of holes" Journal of Fluid Dynamics Research 18 (1996) 237-243.*